Numerical Simulation of Massively Separated flow over Apollo Command Module: Validation Study

Balasubramanyam Sasanapuri, Manish Kumar,
Angela Lestari, Konstantine Kourbatski, Sutikno Wirogo
ANSYS Inc.

Presented By
Reza Ghias

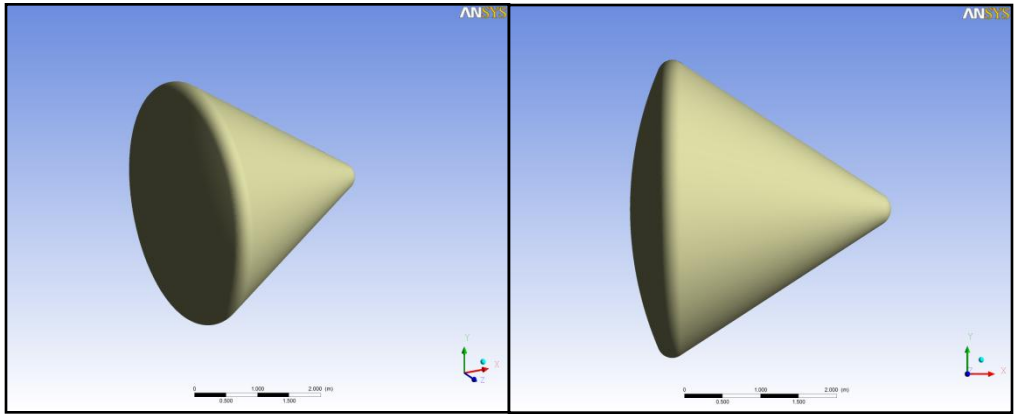
Thermal & Fluids Analysis Workshop
TFAWS 2011
August 15-19, 2011
NASA Langley Research Center
Newport News, VA

Outline

- Objectives
- Model & Flow conditions
- Boundary conditions
- Test cases & Results
- Conclusion

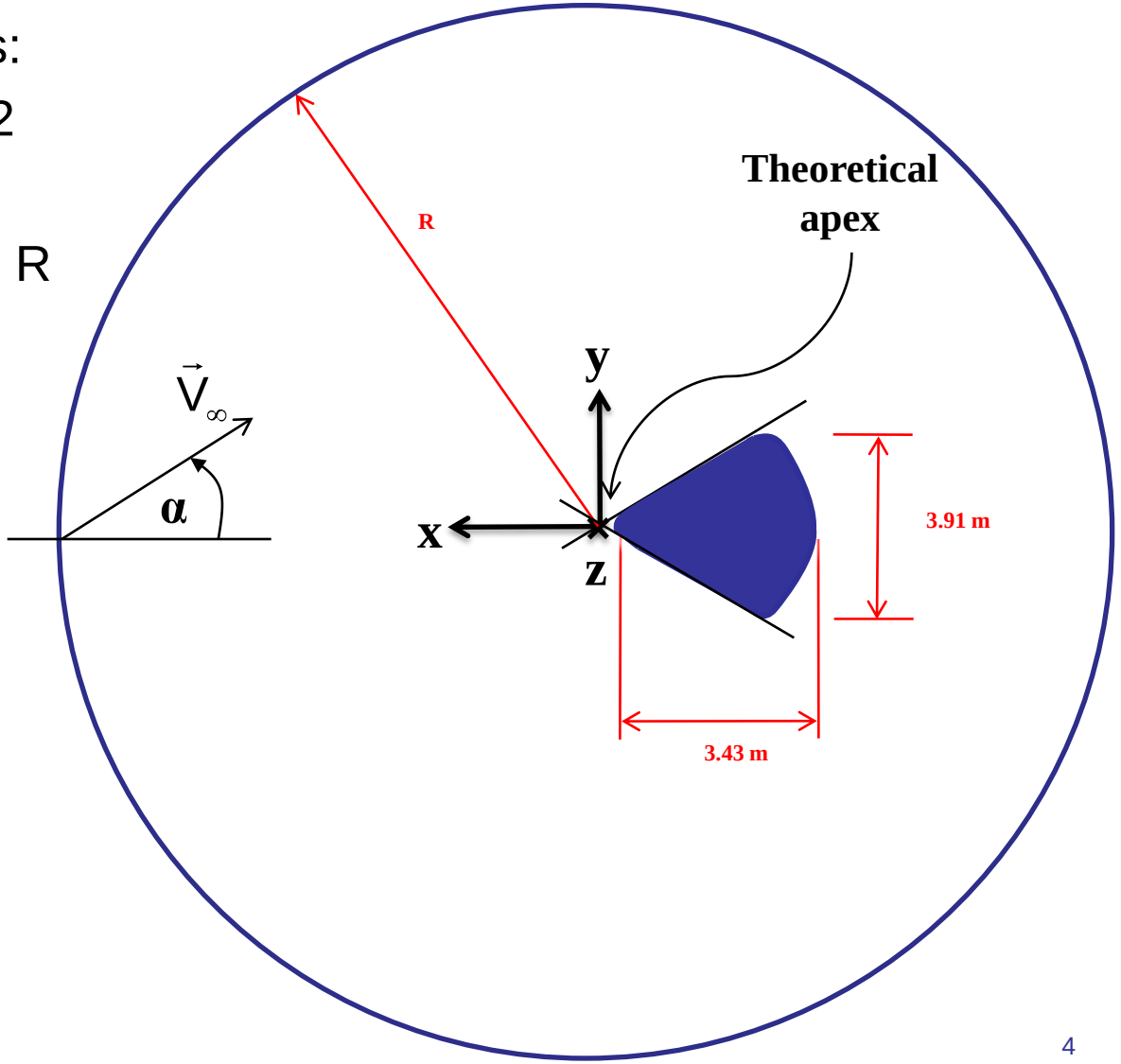
Objectives

- **Problem Statement**
  - Simulate the unsteady separated flow behind the Apollo capsule in supersonic flow
  - Compare predicted force coefficients from ANSYS Fluent simulation with the experimental data (**AIAA 2007-1412**)

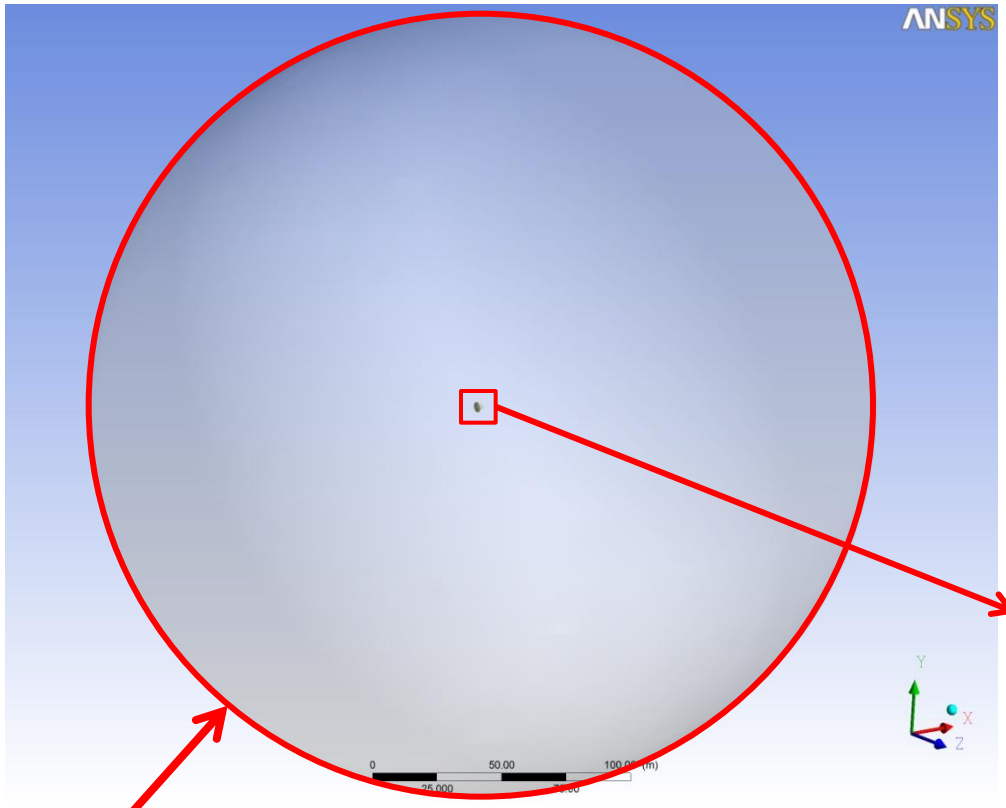


Model and Flow Conditions

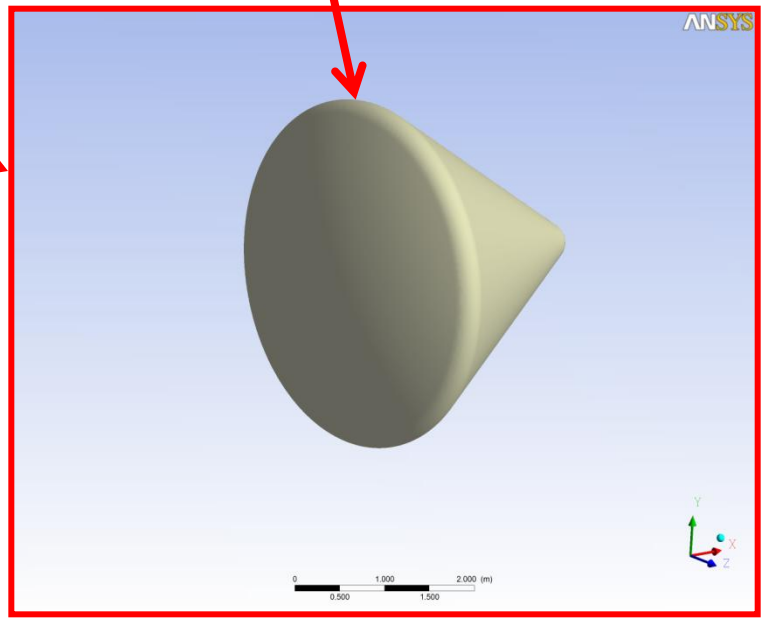
- Free-stream Conditions:
  - Mach Number = 1.2
  - Pressure = 220 Pa
  - Temperature = 420 R



Boundary Conditions



No slip wall



Pressure far-field

P_∞ = 220 Pa

T_∞ = 420 R

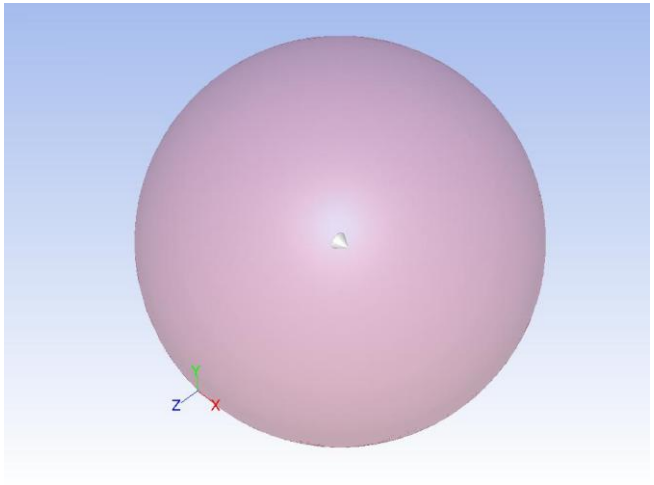
M_∞ = 1.2

Test Cases: Mesh Refinement Study

- **Mesh-0**
  - Coarse Mesh: 4.5 Million Hex Cells
  - Angles of attack studied: 0°, 30°, 60°, 90°, 120°, 150°, 180°
- **Mesh-1**
  - Refined Mesh: 20.5 Million Hex Cells
  - Angle of attack studied: 180°
- **Mesh-2**
  - Refined Mesh: 10.8 Million Cells
    - Boundary layers + Cartesian mesh
    - Refined near shock and wake regions
  - Angles of attack studied: 165°,180°

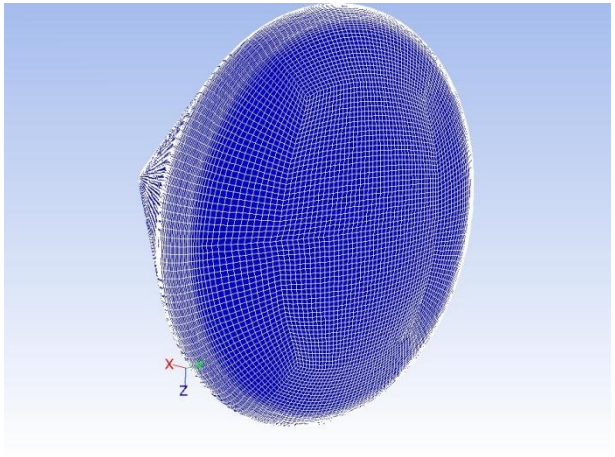
Mesh-0: Description

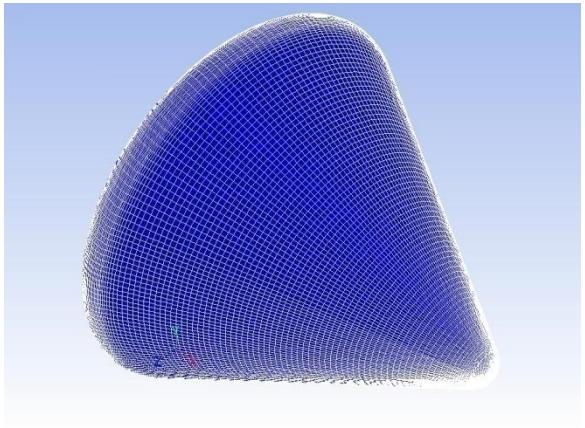
- Hexahedral mesh (4.5 Million cells)
- Outer domain diameter: 22 D
- Wall $Y^+ < 1$
- Angles of attack studied: 0°,30°,60°,90°, 120°,150°,180°



Outer domain

Mesh near the Capsule wall





Surface mesh on the capsule

Mesh-0: Solver Settings

- Solvers
  - Pressure Based Coupled Solver (PBCS) (default)
  - Density Based Navier Stokes (DBNS)
- Turbulence Models
  - Steady: SST k-omega, SST Transition
  - Unsteady: Scale Adaptive Simulation (SAS)
- Discretization
  - Gradients: Least Squares Cell Based
  - Pressure: Second Order (default), PRESTO
  - Momentum, Turbulence, Energy: Second Order

Scale Adaptive Simulation (SAS)

- Uses Von Karman length-scale in the turbulence model to dynamically adjust to the resolved structures in the flow field
- Produces LES-like results for sufficient mesh refinement; otherwise, reverts to RANS
- Unsteady Model

Scale Adaptive Simulation (SAS)

$$\frac{\partial(k)}{\partial t}+\frac{\partial(U_j k)}{\partial x_j}=P_k-c_\mu^{3/4}\frac{k^{3/2}}{L}+\frac{\partial}{\partial x_j}\left(\frac{\nu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right)$$

$$\frac{\partial\Phi}{\partial t}+\frac{\partial(U_j\Phi)}{\partial x_j}=\frac{\Phi}{k}\left(\zeta_1 P_k-\zeta_2\frac{1}{\kappa^2}L^2\nu_t\left(U''\right)^2\right)-\zeta_3\cdot k+\frac{\partial}{\partial y}\left[\frac{\nu_t}{\sigma_\Phi}\frac{\partial\Phi}{\partial y}\right]$$

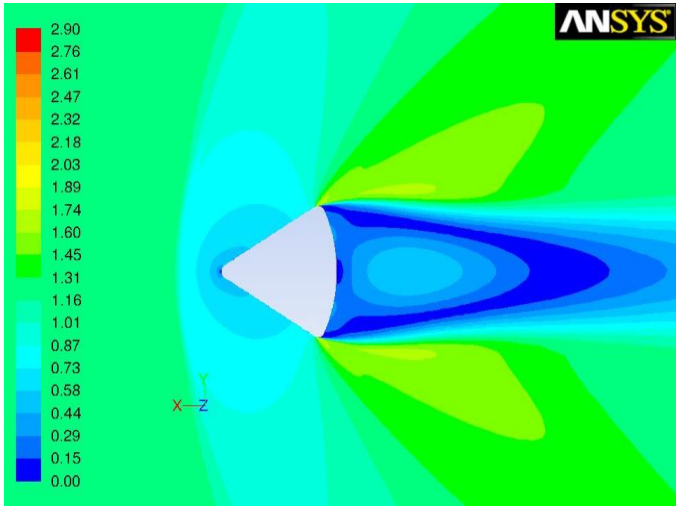
- With:

$$\Phi=\sqrt{k}L \qquad \nu_t=c_\mu^{1/4}\Phi \qquad |U'|=\sqrt{\frac{\partial U_i}{\partial x_j}\frac{\partial U_i}{\partial x_j}}; \quad |U''|=\sqrt{\frac{\partial^2 U_i}{\partial x_j\partial x_j}\frac{\partial^2 U_i}{\partial x_k\partial x_k}}; \quad L_{vK}=\kappa\left|\frac{U'}{U''}\right|$$

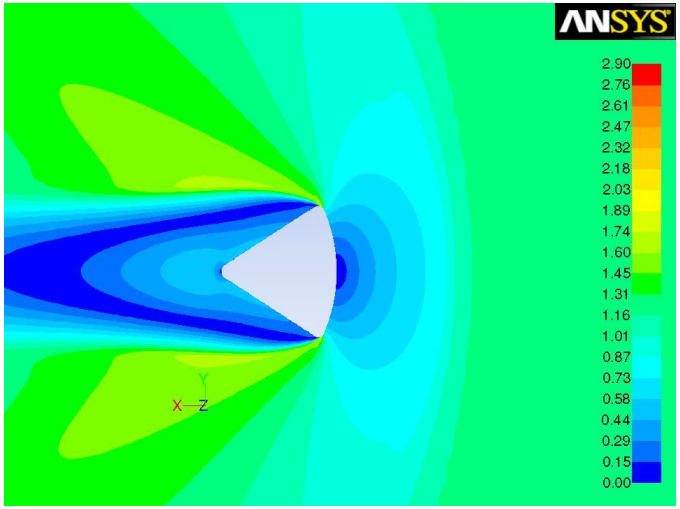
v. Karman length-scale as natural length-scale:

$$L\sim\kappa\left|\frac{\partial U/\partial y}{\partial^2 U/\partial y^2}\right|=L_{vK}$$

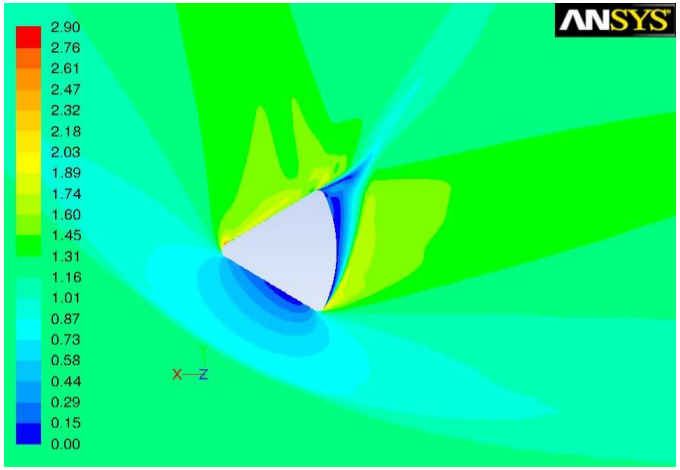
Mesh-0: Results (Mach Contours)



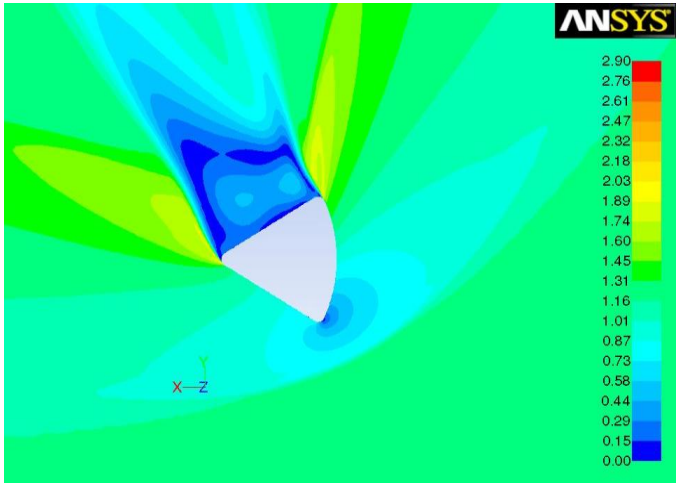
AoA = 0^0



AoA = 180^0

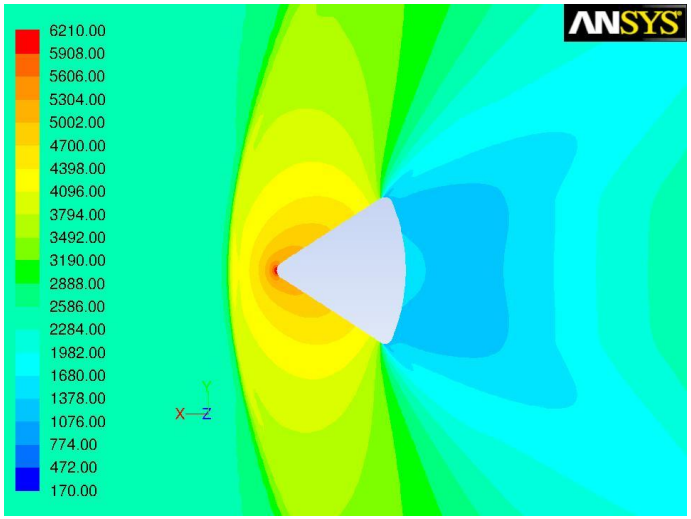


AoA = 60^0

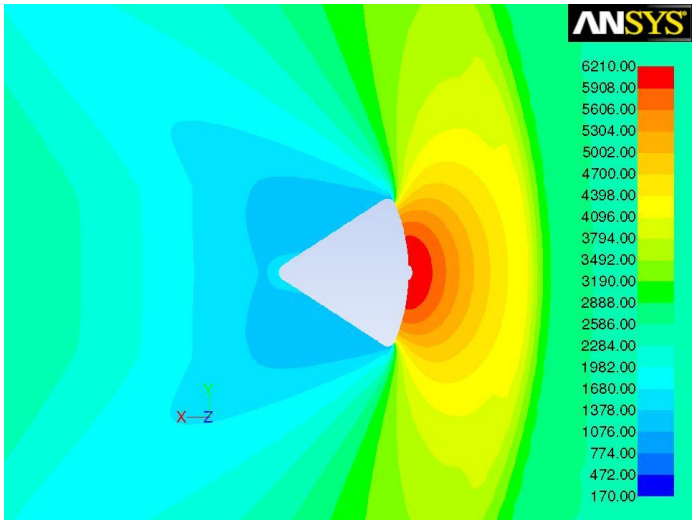


AoA = 120^0

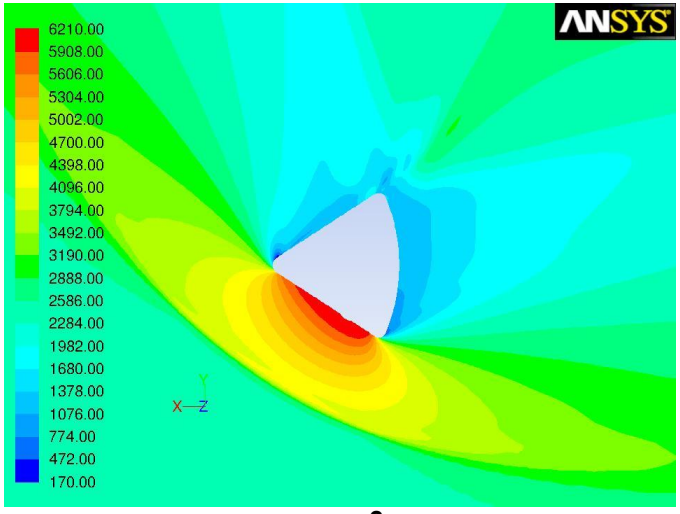
Mesh-0: Results (Pressure Contours)



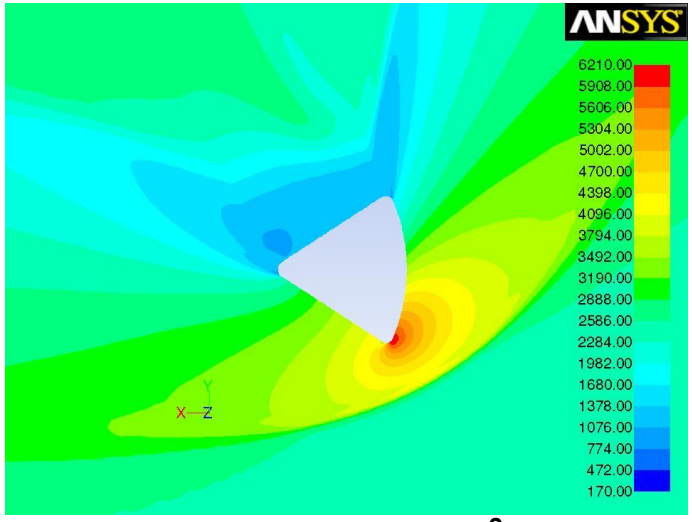
AoA = 0^0



AoA = 180^0

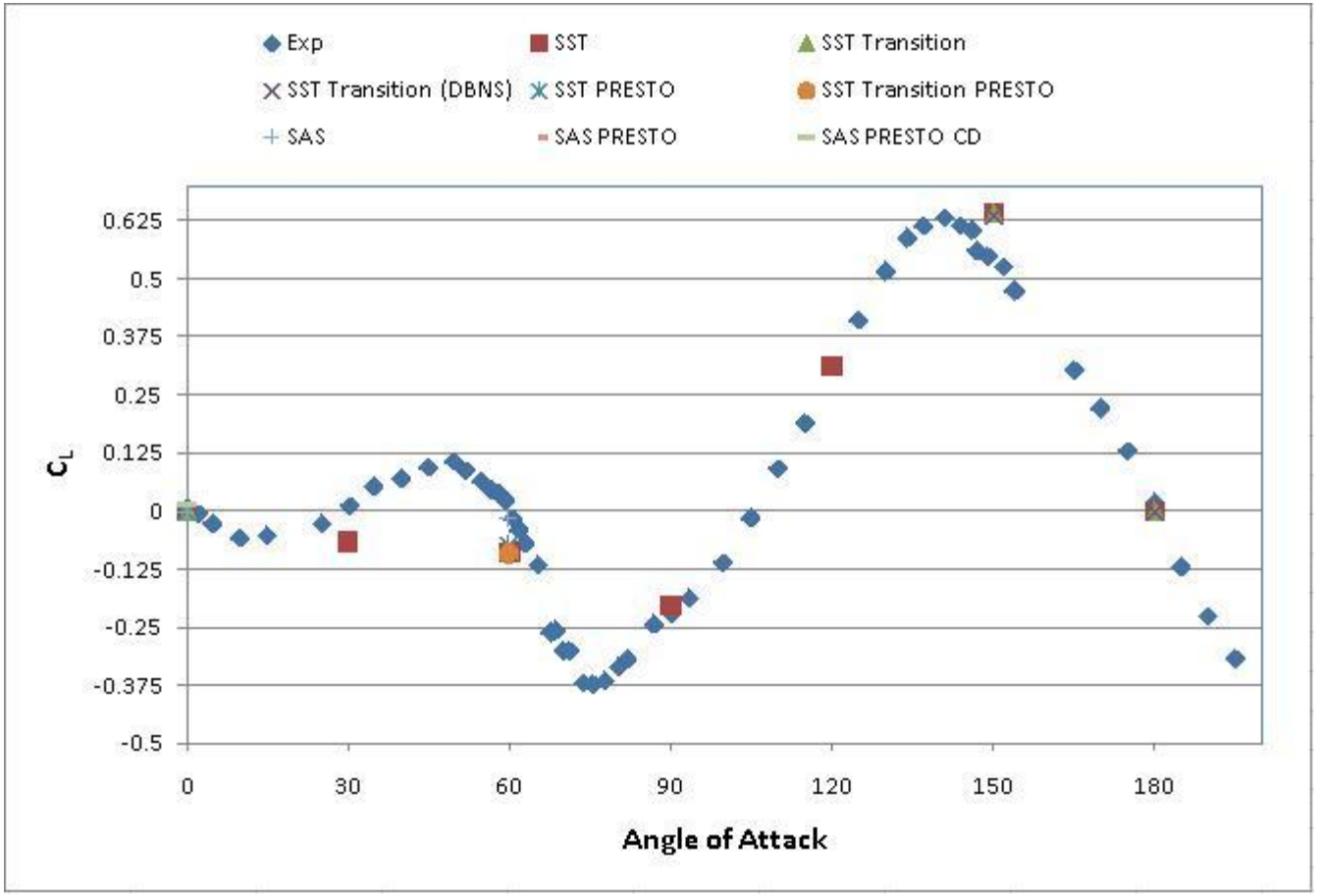


AoA = 60^0

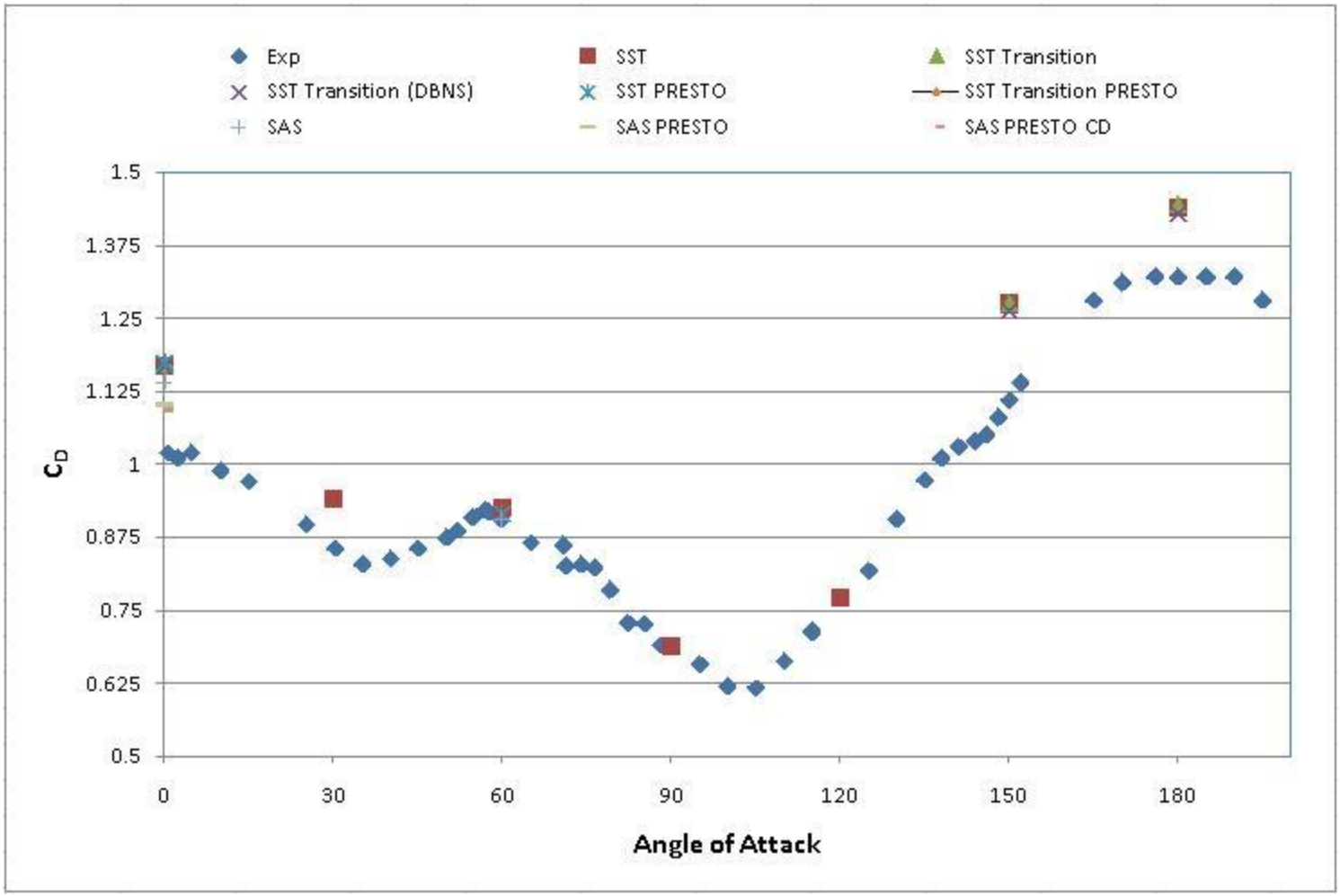


AoA = 120^0

Mesh-0: Results (C_L Plot)



Mesh-0: Results (C_D Plot)

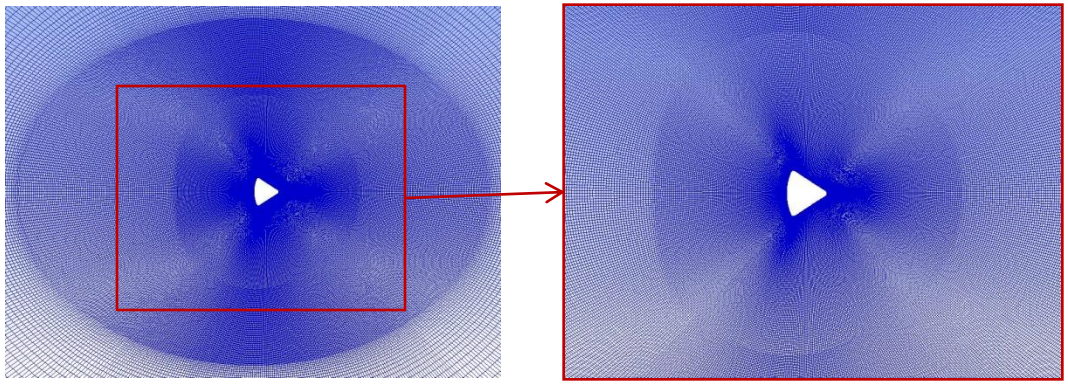


Mesh-0: Summary

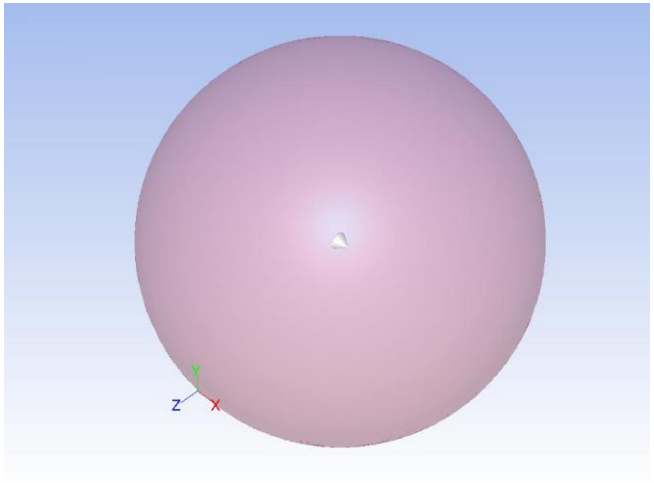
- Initial tests were done for the full AoA range (0 to 180)
- The Force coefficients were predicted well for 60, 90 and 120 AoA but not well for 0, 30, 150 and 180 AoA
- SST Transition model didn’t show improvement over SST k-omega
- Further tests were done with SAS model for 0 AoA
- SAS model has shown significant improvement in solution
- Further improvements are seen with PRESTO scheme for pressure and Central Differencing for momentum
- Mesh resolution (especially in the wake) is not good enough to capture the wake flow

Mesh-1: Description

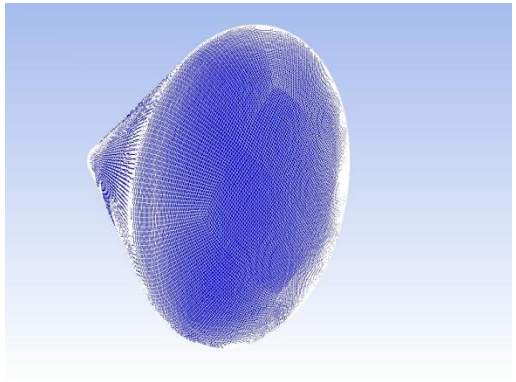
- Hexahedral mesh (20.5 Million cells)
- Outer domain diameter: 76 D
- Wall $Y^+ < 1$
- Angle of attack studied: 180°

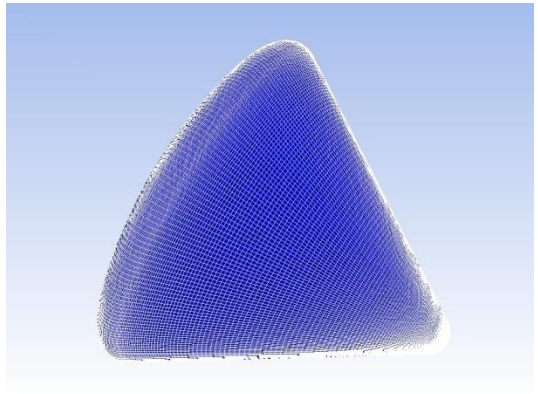


Mesh near the Capsule wall



Outer domain



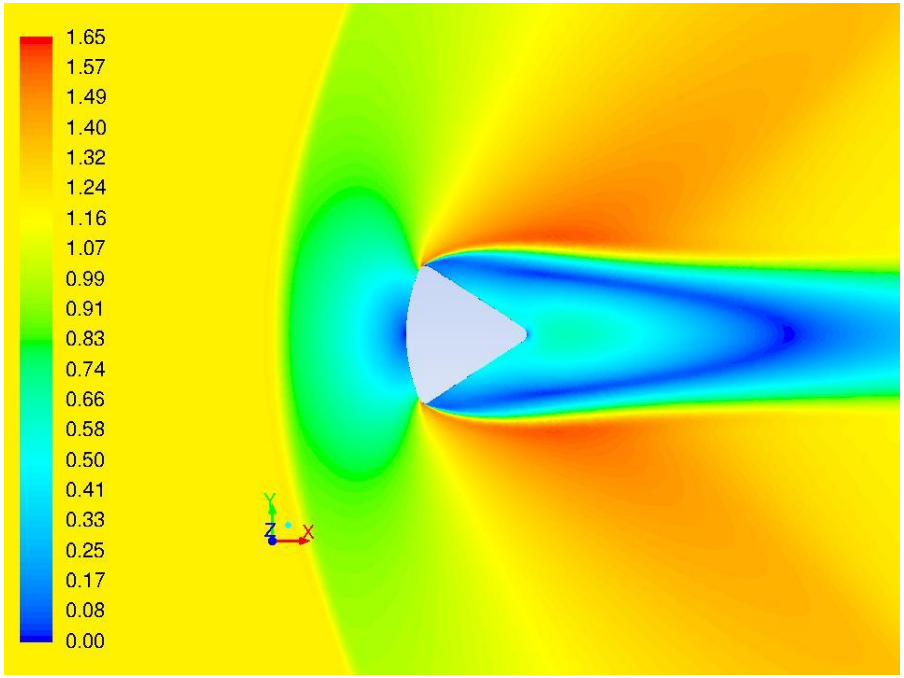


Surface mesh on the capsule

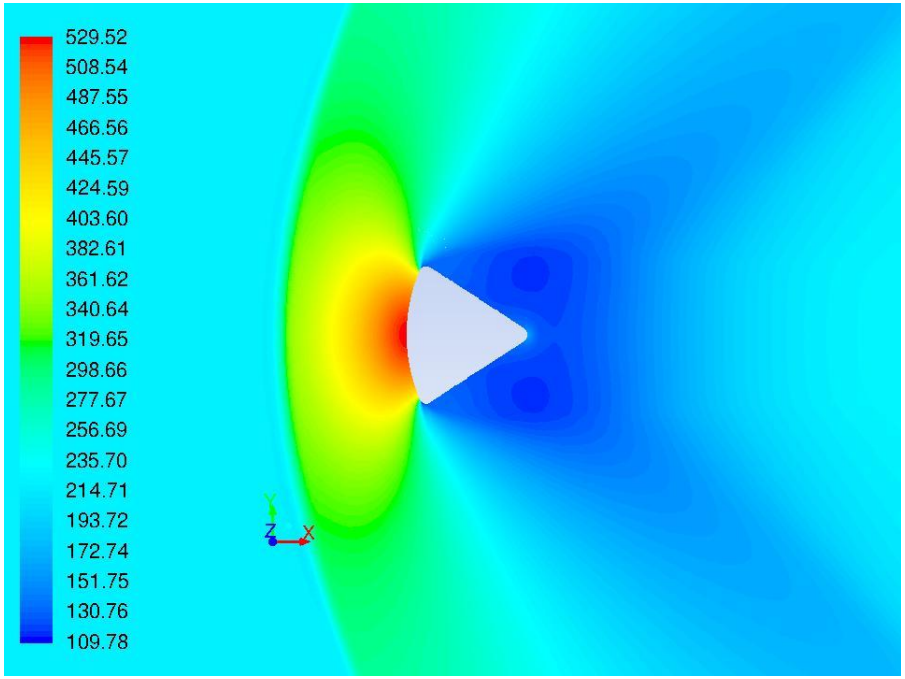
Mesh-1: Solver Settings

- Solvers
  - Pressure Based Coupled Solver (PBCS)
- Turbulence Models
  - Steady: SST k-omega
- Discretization
  - Gradients: Least Squares Cell Based
  - Pressure: PRESTO
  - Momentum, Turbulence, Energy: Second Order

Mesh-1: Results



Mach Contours



Pressure Contours

Comparison: Mesh-0 & Mesh-1

AoA=180		Experimental	SST k-w	Error %
Mesh-0	Lift Coefficient	2.01E-2	2.35E-06	N/A
	Drag Coefficient	1.32	1.4377	8.92%
Mesh-1	Lift Coefficient	2.01E-2	1E-5	N/A
	Drag Coefficient	1.32	1.37	3.79%

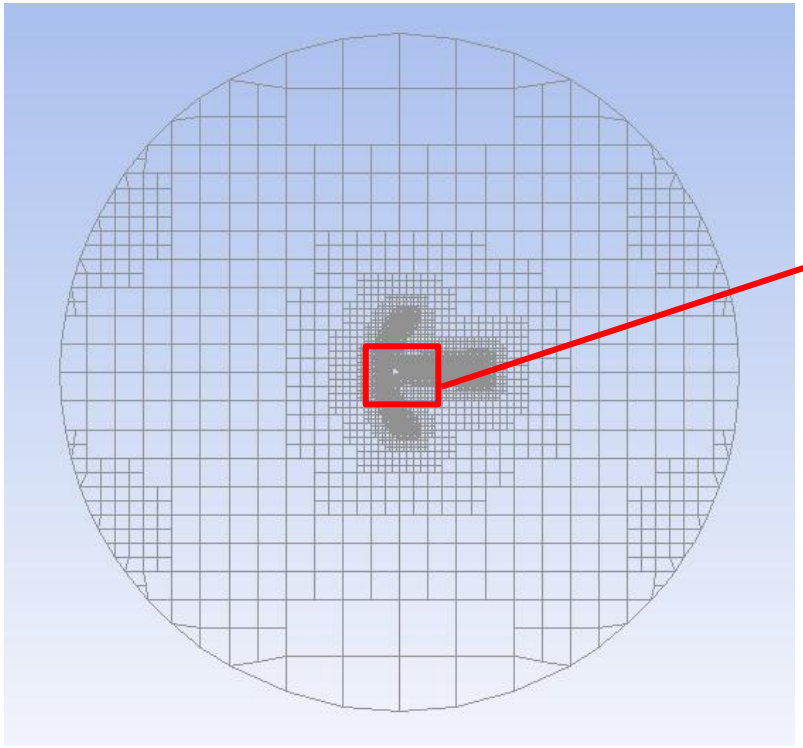
Mesh-1: Summary

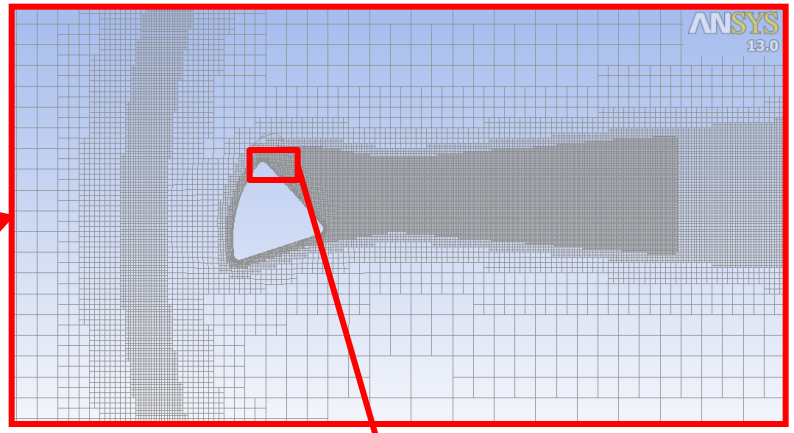
- The refined mesh (Mesh-1) has shown significant (5%) improvement in accuracy over Mesh-0 for AoA 180
- Results are not shown here, but further adaption in the wake didn’t improve the results
- Unsteady SAS simulation on this mesh (20.5 Million) would be quite expensive

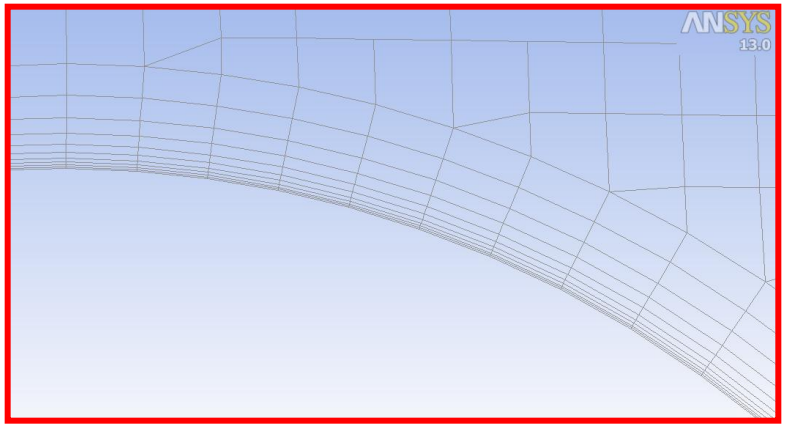
Mesh-2: Objectives

- Reduce the mesh size to around 10 Million Cells
- Cluster the mesh elements where needed (in the initial mesh itself, no adaption)
- Obtain the accuracy comparable to Mesh-1 (fine mesh)
- AoA Studied: 165, 180

Mesh-2: Details (AoA=165°)

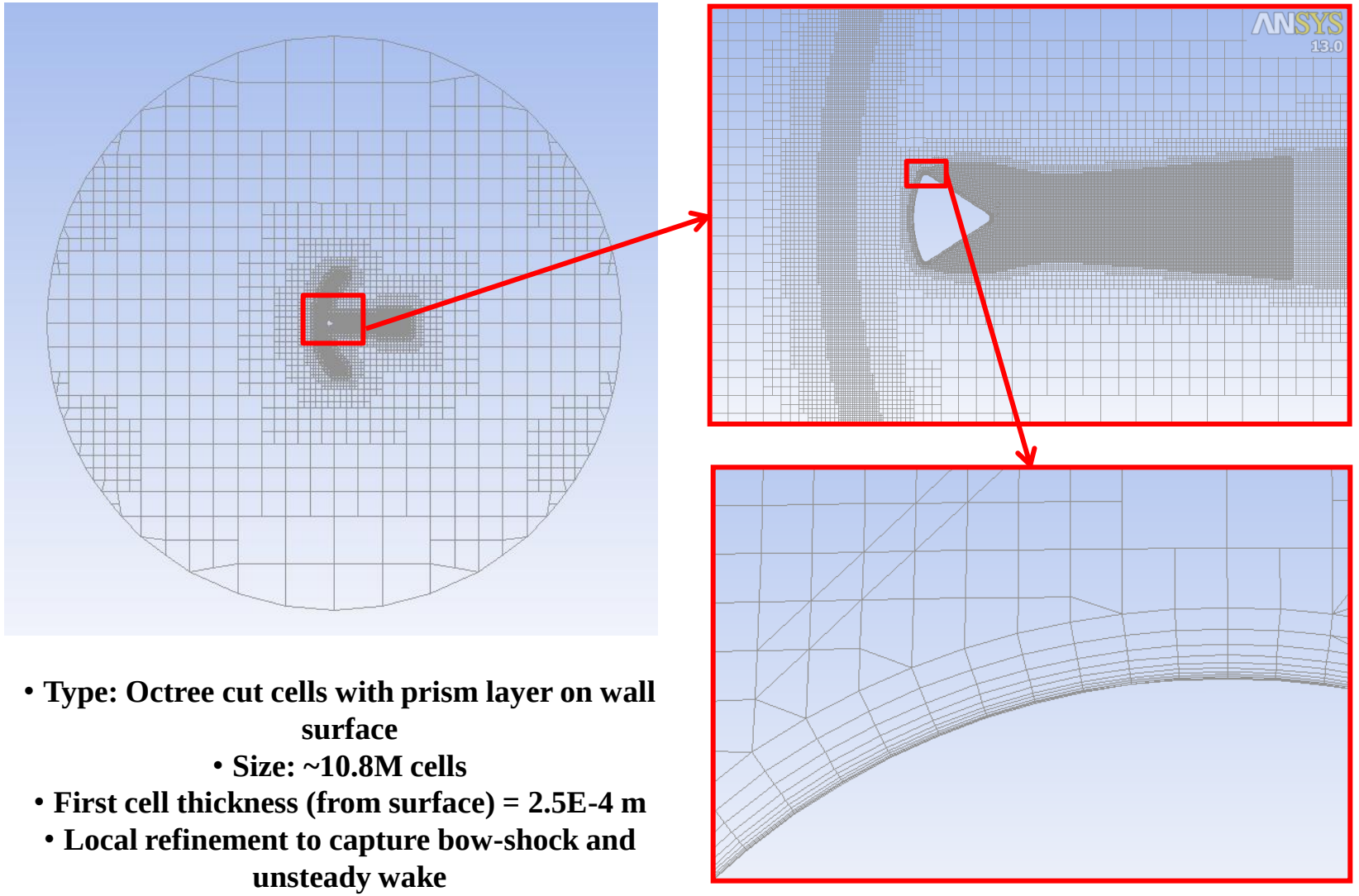






- **Type: Octree cut cells with prism layer on wall surface**
- **Size: ~10.8M cells**
- **First cell thickness (from surface) = 2.5E-4 m**
- **Local refinement to capture bow-shock and unsteady wake**

Mesh-2: Details (AoA=180°)



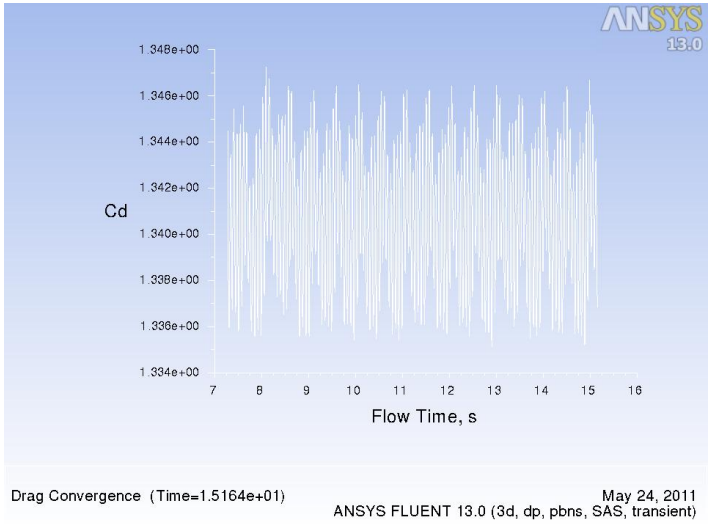
- Type: Octree cut cells with prism layer on wall surface
- Size: ~10.8M cells
- First cell thickness (from surface) = 2.5E-4 m
- Local refinement to capture bow-shock and unsteady wake

Mesh-2: Solver Settings

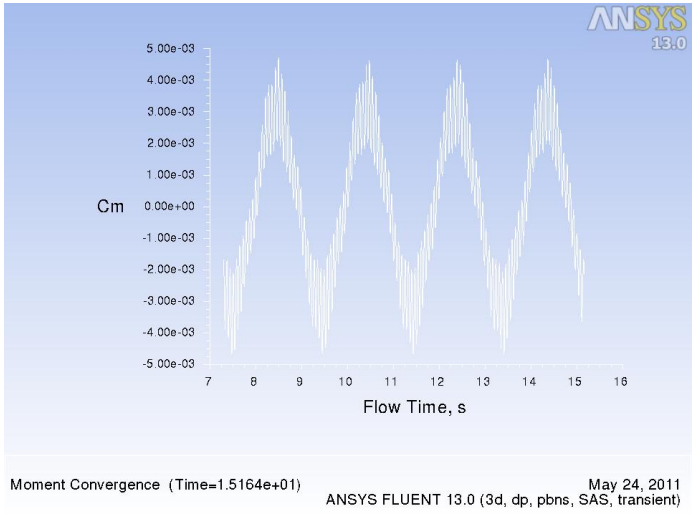
- FLUENT Pressure-Based Navier-Stokes Solver
- Spatial Discretization
  - PRESTO for pressure
  - Bounded Central Differencing for momentum
  - 2nd order Upwind for other equations
- SAS Turbulence Model
- Transient Solver
  - Second Order Implicit
  - Δt = 0.005 second for AoA of 180°, 0.01 second for AoA of 165°
  - 20 iterations per time-step

Mesh-2: AoA 180

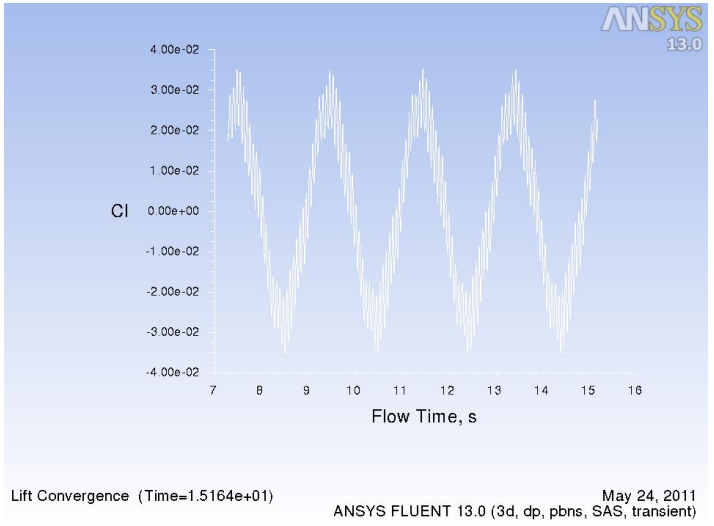
- Simulation is run until periodic behavior is seen
- Time averaged quantities are obtained for comparison



Drag Coefficient history

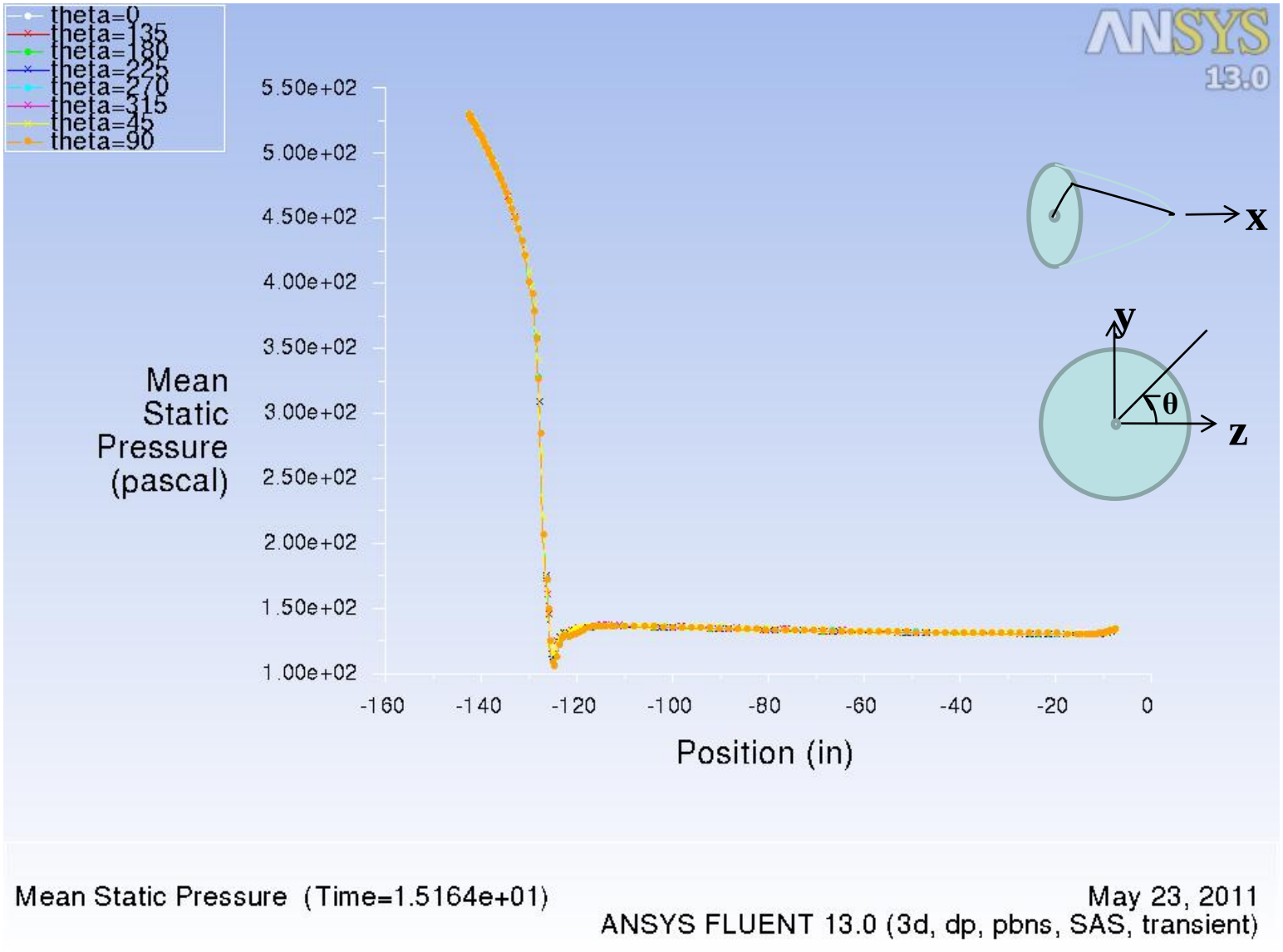


Moment Coefficient history

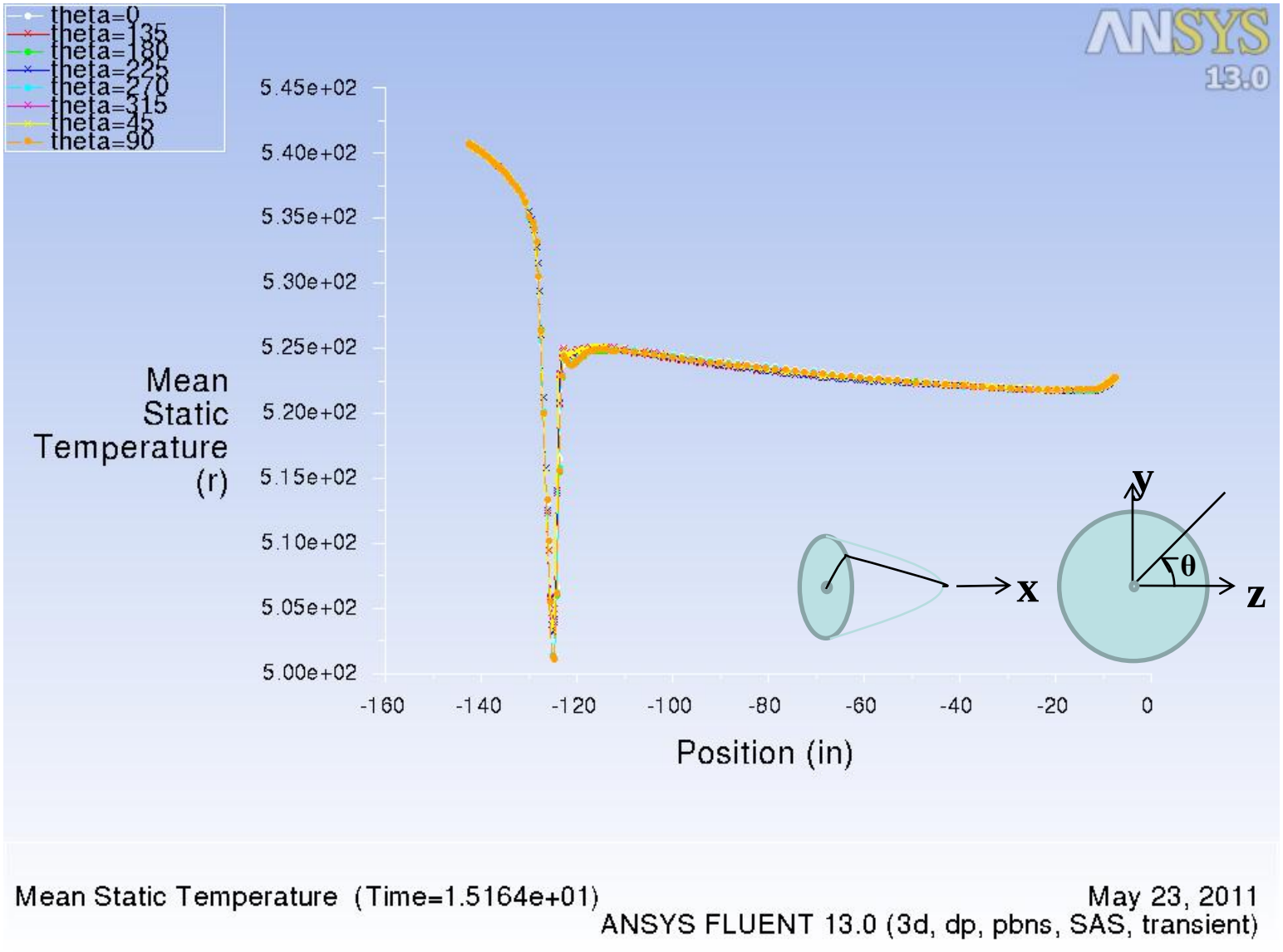


Lift Coefficient history

Time-averaged Surface Pressure

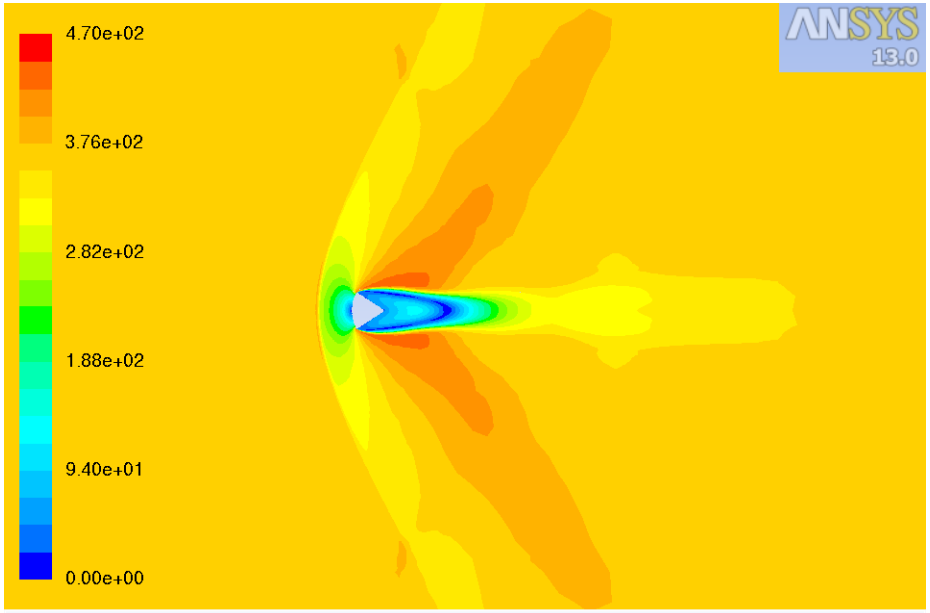


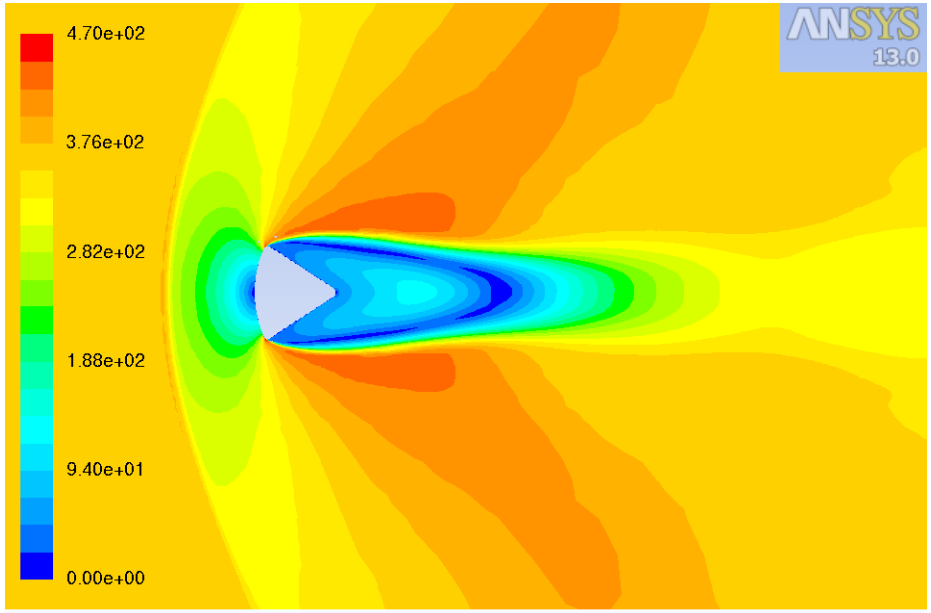
Time-averaged Surface Temperature



Time-averaged Velocity Field

Plane z=0

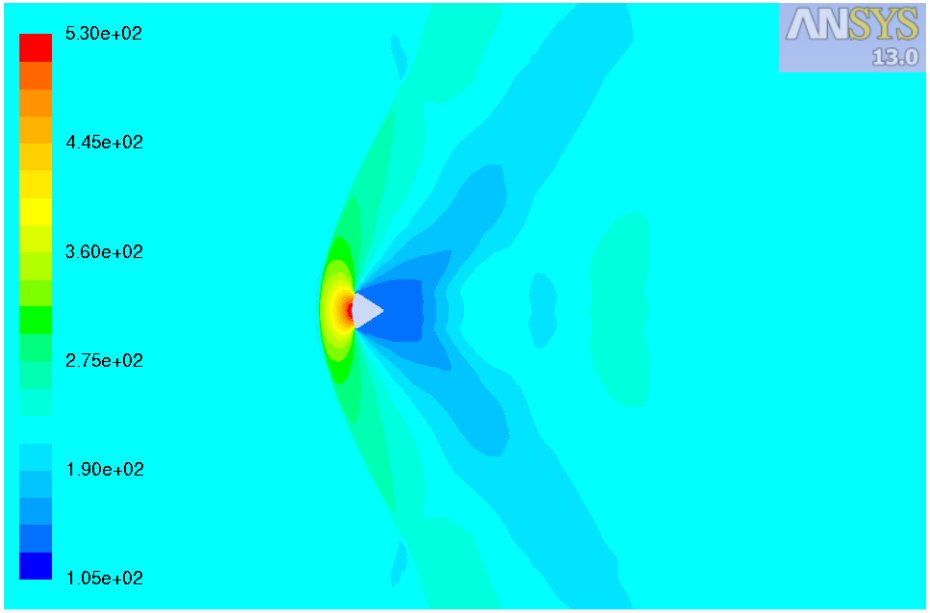


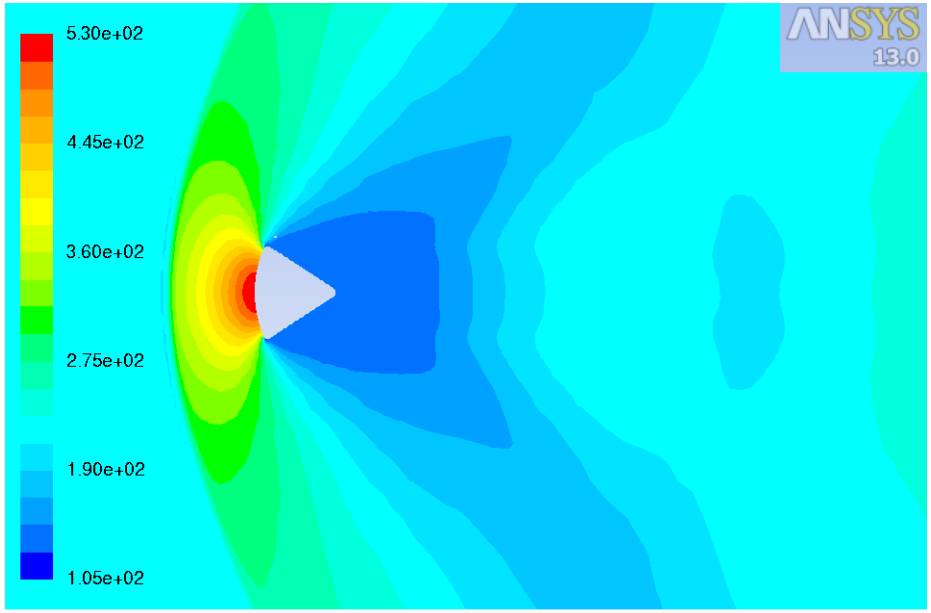


Zoomed-in View

Time-averaged Pressure Field

Plane z=0

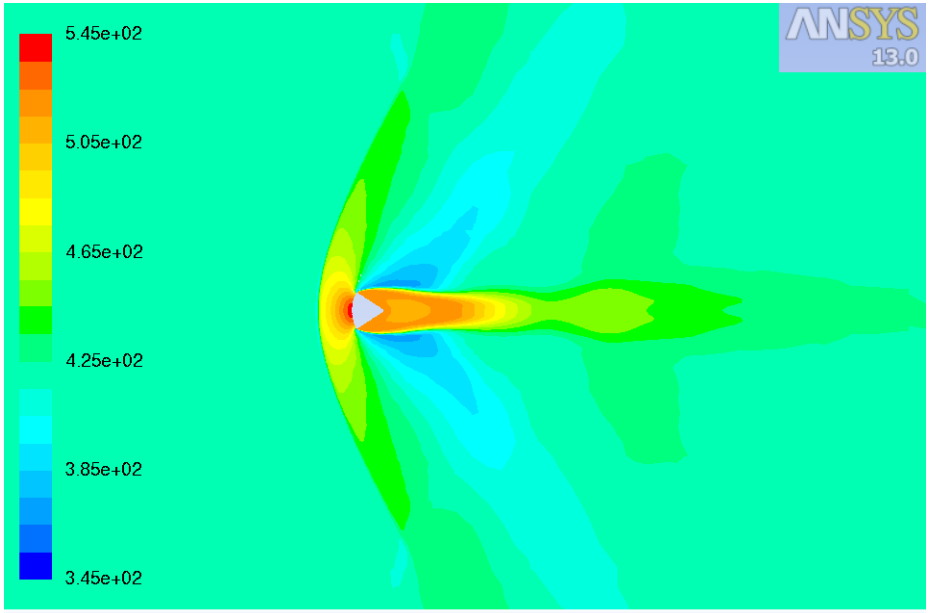




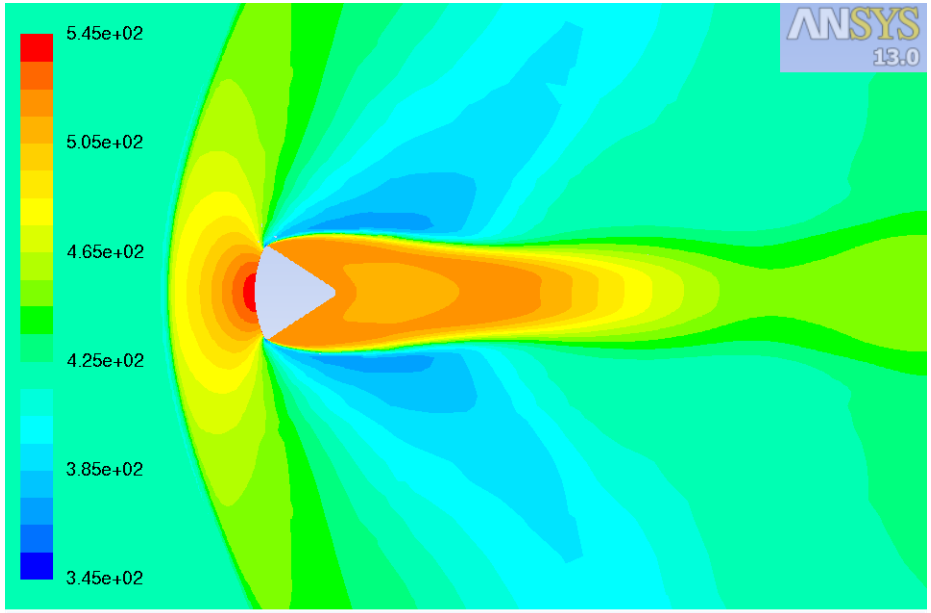
Zoomed-in View

Time-averaged Temperature Field

Plane z=0



Contours of Mean Static Temperature (r) (Time=1.5164e+01) May 24, 2011
ANSYS FLUENT 13.0 (3d, dp, pbns, SAS, transient)

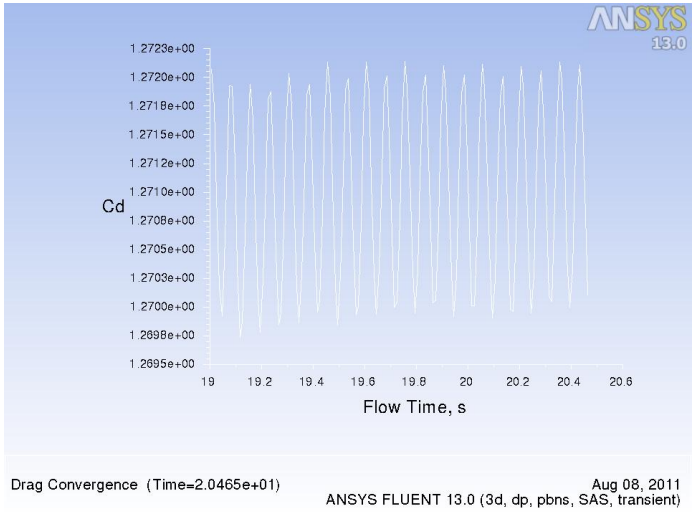


Contours of Mean Static Temperature (r) (Time=1.5164e+01) May 24, 2011
ANSYS FLUENT 13.0 (3d, dp, pbns, SAS, transient)

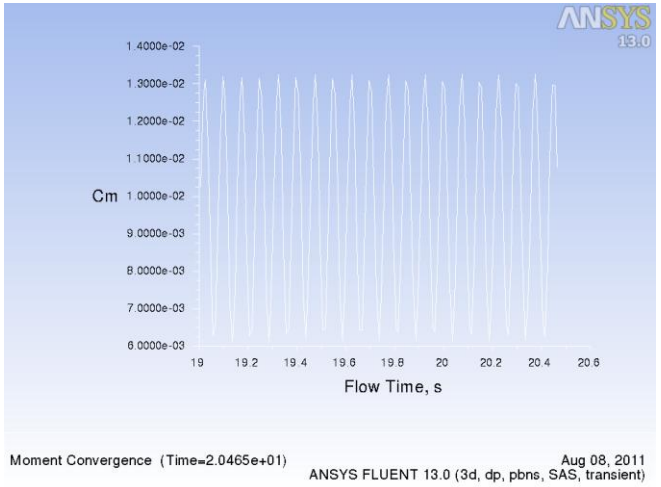
Zoomed-in View

Mesh-2: AoA 165

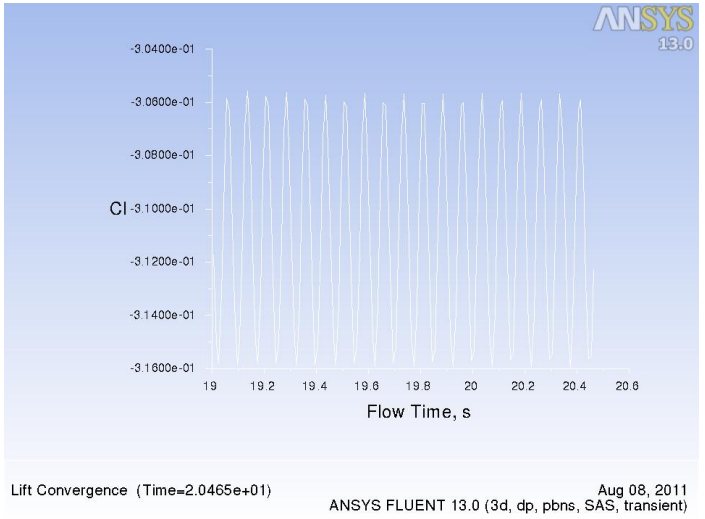
- Simulation is run until periodic behavior is seen
- Time averaged quantities are obtained for comparison



Drag Coefficient history



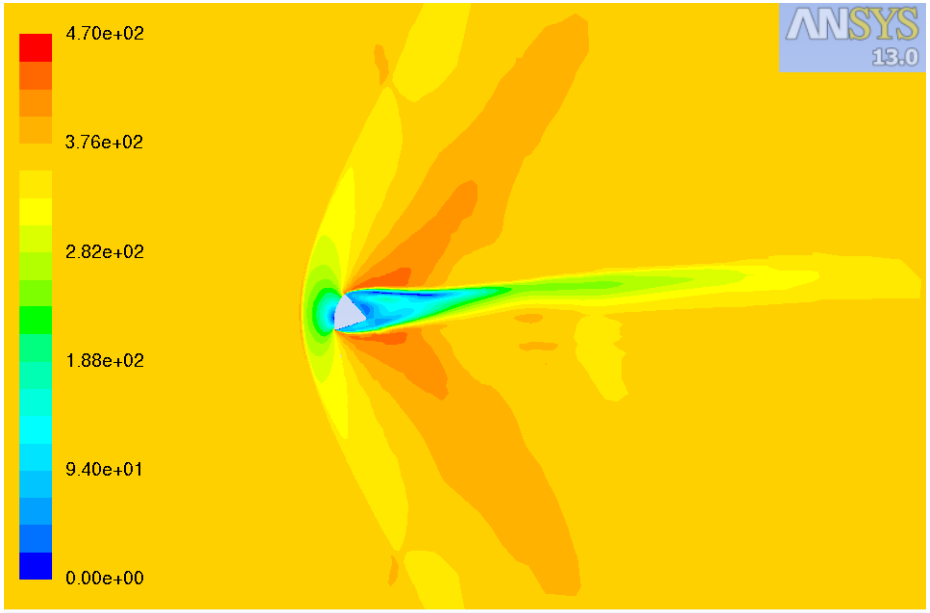
Moment Coefficient history

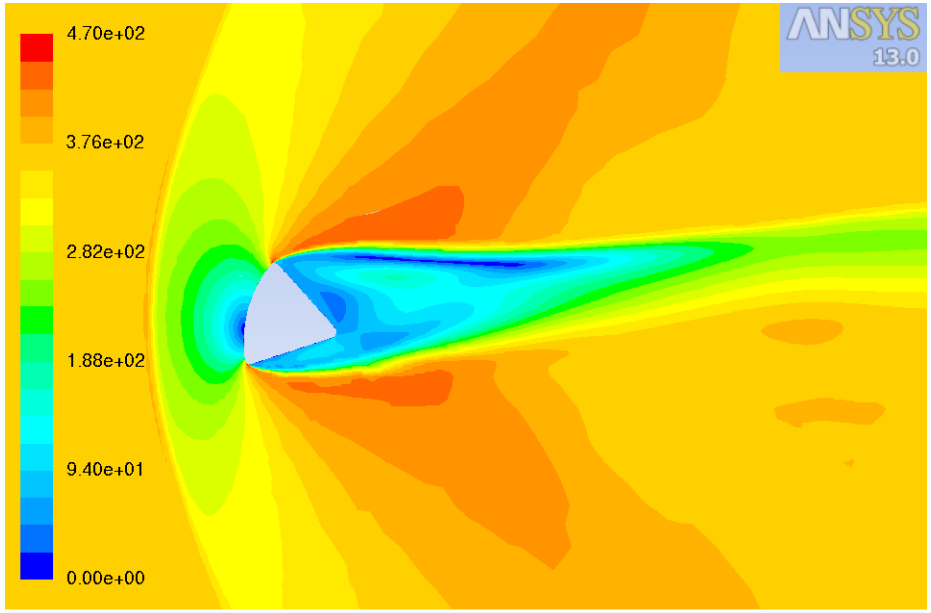


Lift Coefficient history

Time-averaged Velocity Field

Plane z=0

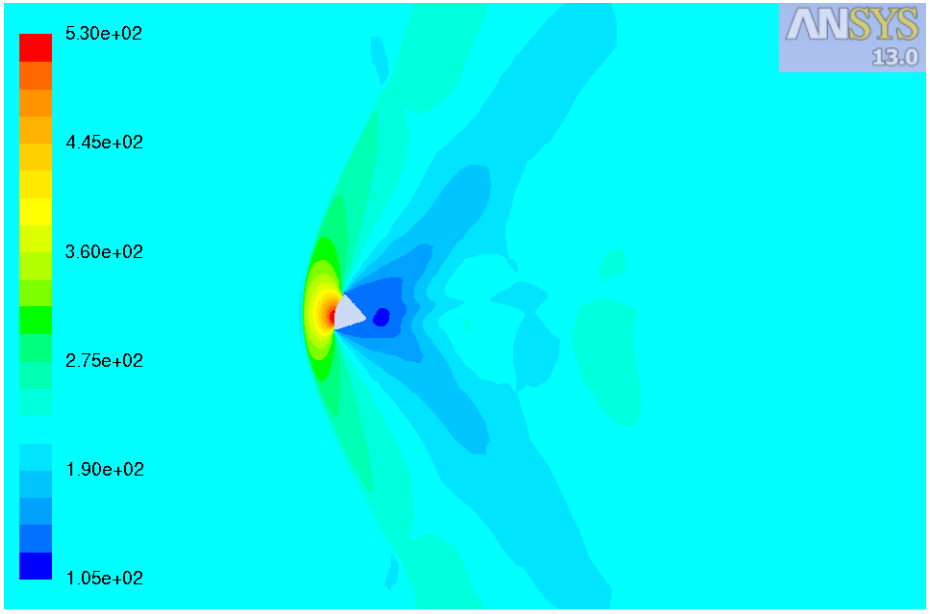




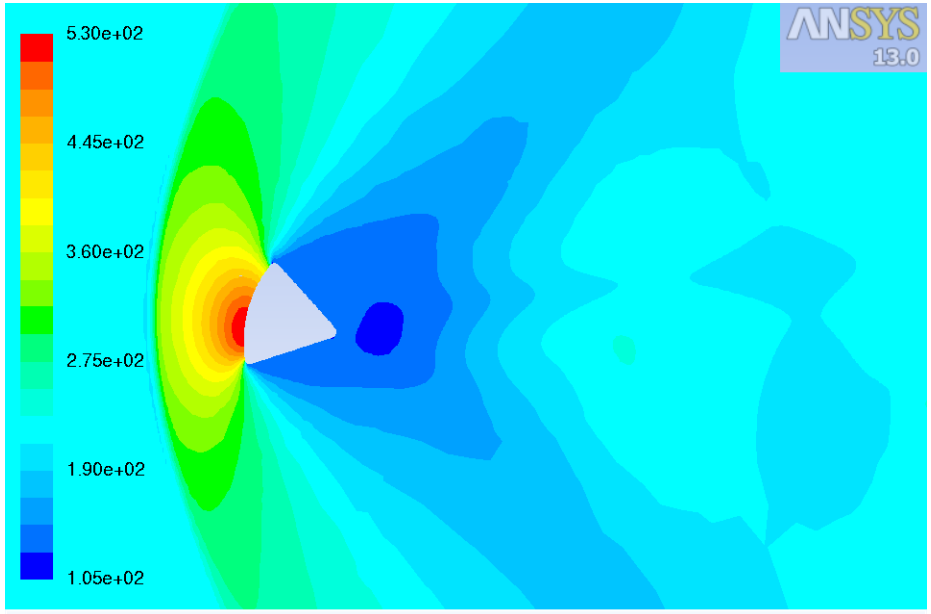
Zoomed-in View

Time-averaged Pressure Field

Plane z=0



Contours of Mean Static Pressure (pascal) (Time=2.0465e+01) Aug 05, 2011
ANSYS FLUENT 13.0 (3d, dp, pbns, SAS, transient)

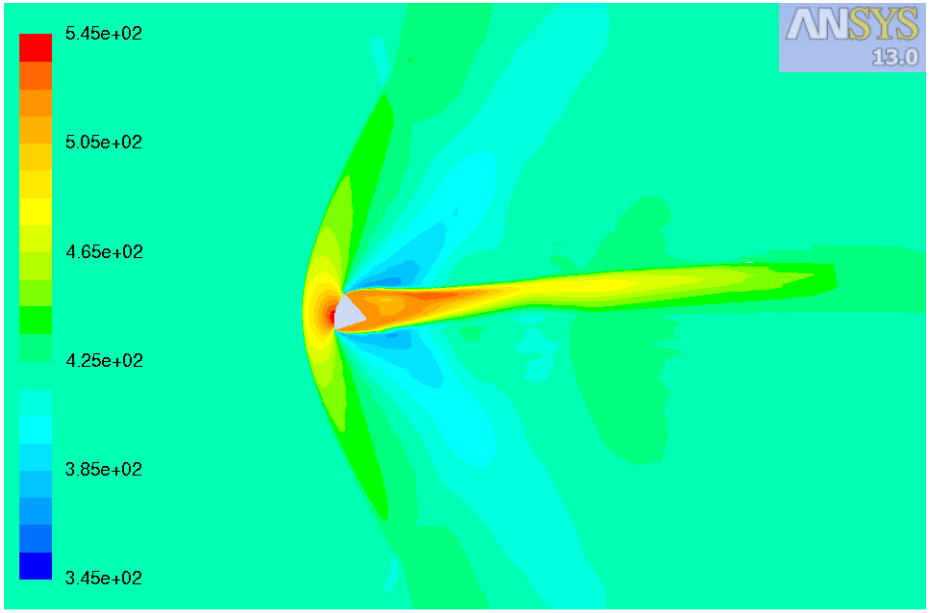


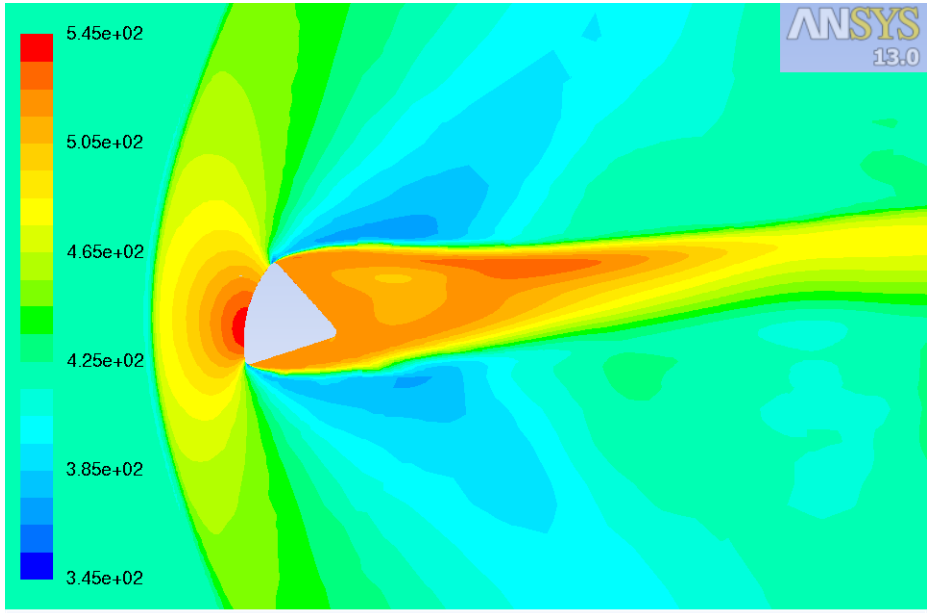
Contours of Mean Static Pressure (pascal) (Time=2.0465e+01) Aug 05, 2011
ANSYS FLUENT 13.0 (3d, dp, pbns, SAS, transient)

Zoomed-in View

Time-averaged Temperature Field

Plane z=0





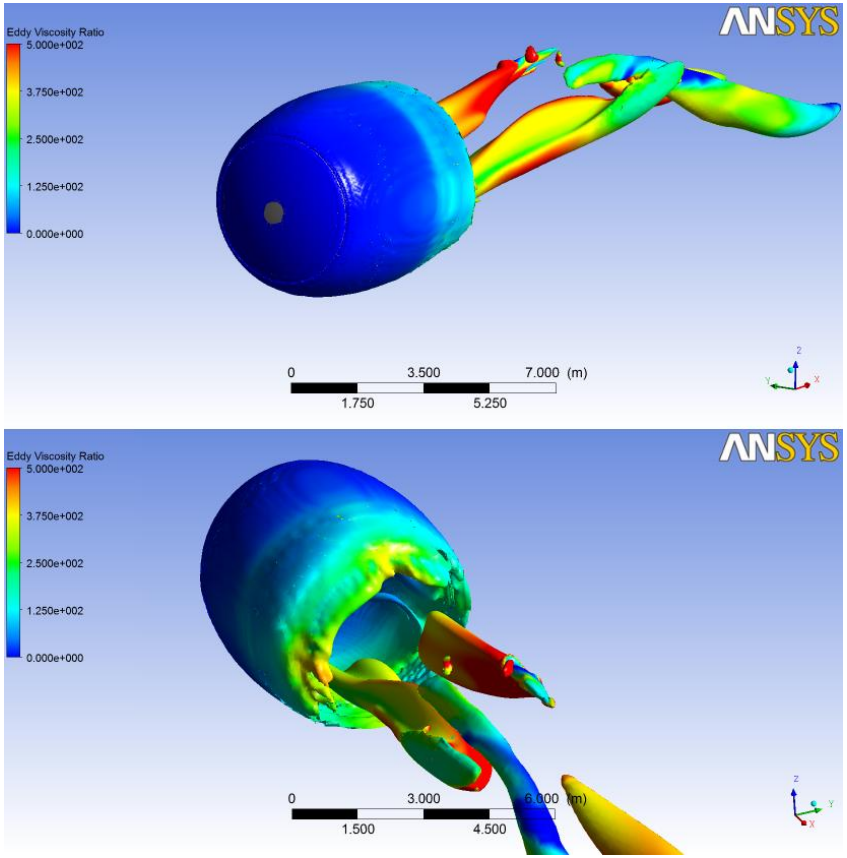
Zoomed-in View

Force and Moment Comparison

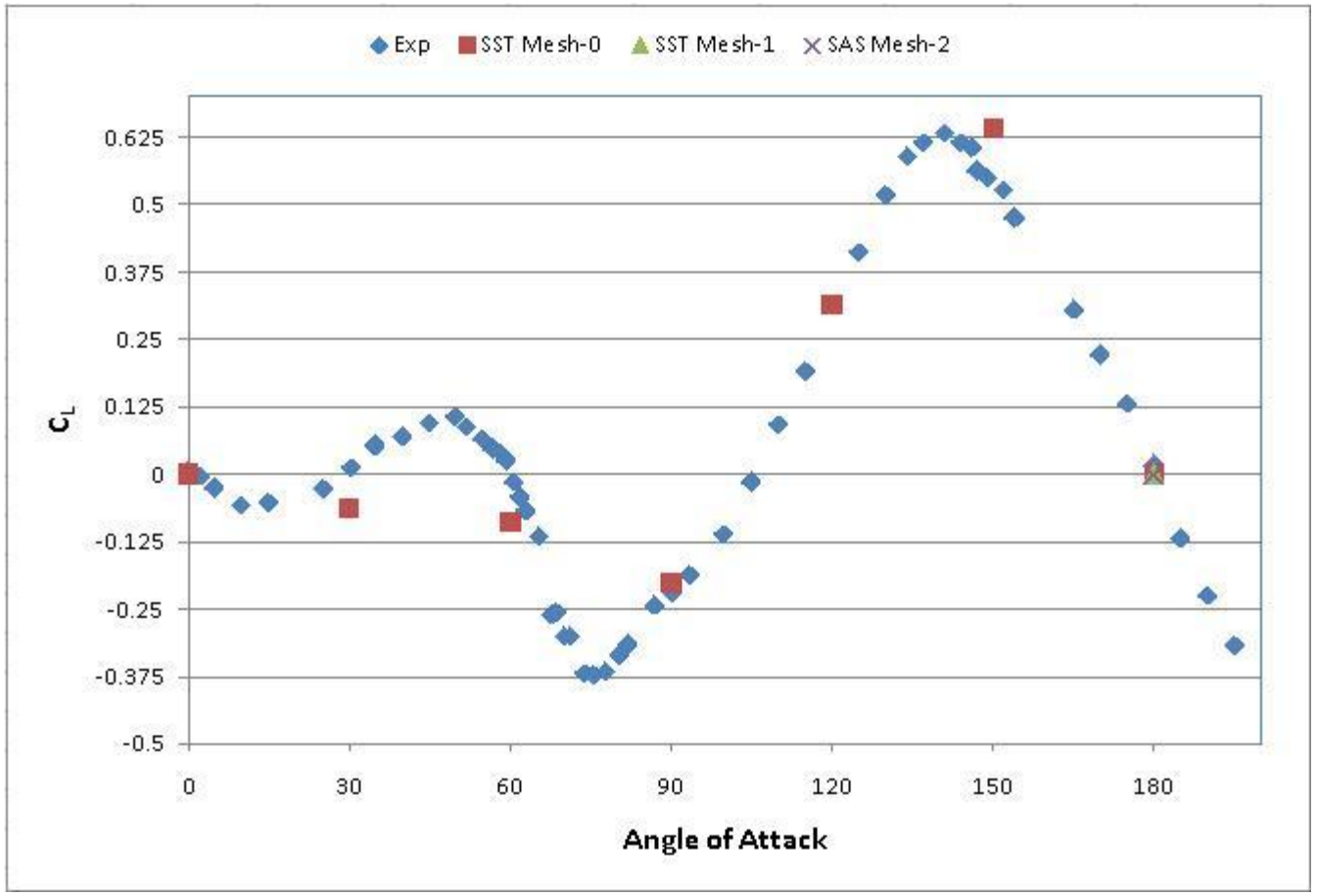
		Experimental	Time-averaged Unsteady SAS	Error %
AoA=180	Lift Coefficient	2.01E-2	5.93E-05	N/A
	Drag Coefficient	1.32	1.3407	1.57%
	Moment Coefficient	N/A	4.26E-06	N/A
AoA=165	Lift Coefficient	0.3	0.310892	3.63%
	Drag Coefficient	1.275	1.27118	0.3%
	Moment Coefficient	N/A	0.0097351	N/A

SAS turbulence model effect(AoA = 180°)

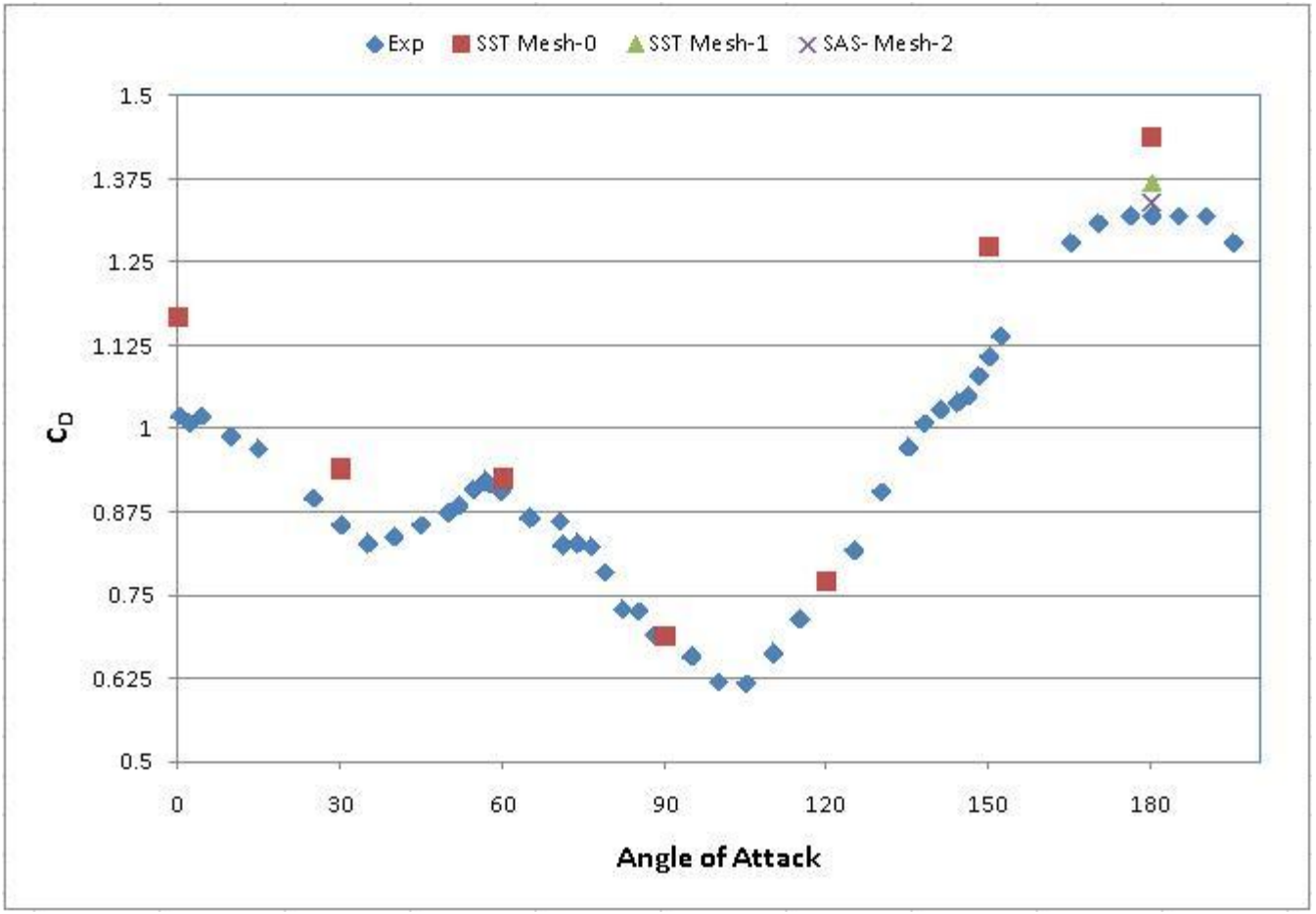
Unsteady SAS



Mesh comparison: Cl



Mesh comparison: Cd



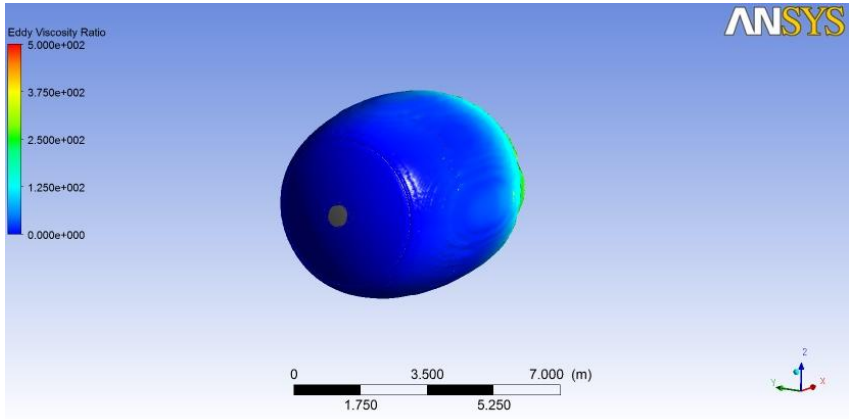
Conclusion

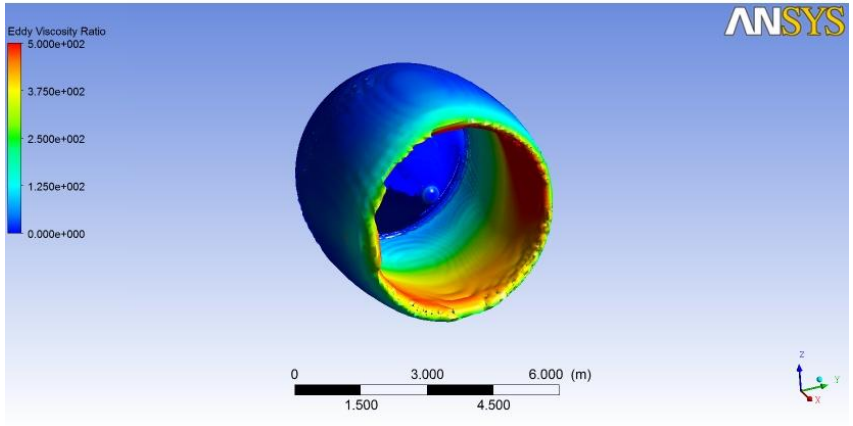
- ANSYS Prism Layer and Octree based cut-cell technology together proved to be powerful and cost-effective
  - The 10.8 Million mesh could achieve better results compared to 20.5 Million hex mesh
- ANSYS FLUENT with transient SAS turbulence model accurately captured the unsteady vortex shedding phenomena behind the Apollo Capsule
- The accuracy of drag coefficient prediction is within 1.57% of the experimental data

APPENDIX

Turbulence Model Comparison (AoA = 180°)

SST k-ω





Unsteady SAS